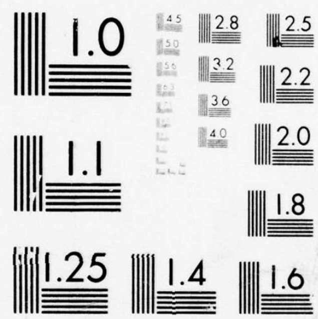
AD-A071 049 DCW INDUSTRIES INC STUDIO CITY CA F/G 20/4
DEVELOPMENT OF A VISCOUS THEORY OF THREE-DIMENSIONAL FLOW IN HI--ETC(U)
JUN 79 D C WILCOX F49620-78-C-0024
UNCLASSIFIED DCW-R-21-01 AFOSR-TR-79-0804 NL

1 OF 1
AD
A071049

END
DATE
FILMED
8--79
DDC



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

LEVEL (4) B.S.

DCW-R-21-01

DEVELOPMENT OF A VISCOUS THEORY OF THREE-DIMENSIONAL FLOW IN HIGHLY-LOADED AXIAL TURBOMACHINES

by

David C. Wilcox

INTERIM SCIENTIFIC REPORT
June 1979

Prepared for

AIR FORCE OFFICE OF SCIENTIFIC RESEARCH
Air Force Systems Command
Bolling AFB, Washington, D.C.

Under Contract F49620-78-C-0024

DCW Industries, Inc.
4367 Troost Avenue
Studio City, California 91604
213/769-7571

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

REPORT DOCUMENTATION PAGE		READ INSTRUCTIONS BEFORE COMPLETING FORM
1. REPORT NUMBER AFOSR-TR- 79-0804	2. GOVT ACCESSION NO.	3. RECIPIENT'S CATALOG NUMBER
4. TITLE (and Subtitle) DEVELOPMENT OF A VISCOUS THEORY OF THREE-DIMENSIONAL FLOW IN HIGHLY-LOADED AXIAL TURBOMACHINES		5. TYPE OF REPORT & PERIOD COVERED INTERIM 31 Mar 78 - 31 Mar 79
		6. PERFORMING ORG. REPORT NUMBER
7. AUTHOR(s) DAVID C WILCOX		8. CONTRACT OR GRANT NUMBER(s) F49620-78-C-0024
9. PERFORMING ORGANIZATION NAME AND ADDRESS DCW INDUSTRIES, INC 4367 TROOST AVE STUDIO CITY, CALIFORNIA 91604		10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS 2037A3 61102F
11. CONTROLLING OFFICE NAME AND ADDRESS AIR FORCE OFFICE OF SCIENTIFIC RESEARCH/NA BLDG 410 BOLLING AIR FORCE BASE, D C 20332		12. REPORT DATE June 1979
		13. NUMBER OF PAGES 29
14. MONITORING AGENCY NAME & ADDRESS(if different from Controlling Office)		15. SECURITY CLASS. (of this report) UNCLASSIFIED
		15a. DECLASSIFICATION/DOWNGRADING SCHEDULE

16. DISTRIBUTION STATEMENT (of this Report)

Approved for public release; distribution unlimited.

17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)

18. SUPPLEMENTARY NOTES

19. KEY WORDS (Continue on reverse side if necessary and identify by block number)

THREE-DIMENSIONAL VISCOUS FLOW
NUMERICAL SIMULATION
ROTATING TURBOMACHINES

20. ABSTRACT (Continue on reverse side if necessary and identify by block number)

A computer program is being developed which will be suitable for numerically simulating three-dimensional viscous flow in rotating turbomachines. This report summarizes progress to date and gives projections of work to be accomplished during the next year.

DD FORM 1 JAN 73 1473

UNCLASSIFIED

14 DCW-R-21-Ø1

6

DEVELOPMENT OF A VISCOUS THEORY OF THREE-DIMENSIONAL FLOW IN HIGHLY-LOADED AXIAL TURBOMACHINES.

by

10 David C. Wilcox

9 INTERIM SCIENTIFIC REPORT. 31 Mar 78 - 31 Mar 79

11 June 1979

12 31p.

Prepared for

AIR FORCE OFFICE OF SCIENTIFIC RESEARCH
Air Force Systems Command
Bolling AFB, Washington, D.C.

15 Under Contract F49620-78-C-0024

16 2137 17 A3

18 AFOSR

DCW Industries, Inc.
4367 Troost Avenue
Studio City, Ca~~lifornia~~ 91604
213/769-7571

19 TR-79-0804

393 352

LB

FOREWORD

This report summarizes research performed in Contract F49620-78-C-0024 during the period March 31, 1978 through March 30, 1979. This research was sponsored by the Air Force Office of Scientific Research (AFSC), United States Air Force. The Air Force program monitors were Lt. Col. Robert Smith and Dr. George Samaras.

Study participants were Dr. David Wilcox, principal investigator, and B. A. Wilcox, data processing support. Mr. Robert MacCormack of the NASA Ames Research Center and Dr. James McCune of the Massachusetts Institute of Technology provided invaluable consulting assistance. Manuscript preparation was accomplished by Ms. Lottie Pilner.

ABSTRACT

A computer program is being developed which will be suitable for numerically simulating three-dimensional viscous flow in rotating turbomachines. This report summarizes progress to date and gives projections of work to be accomplished during the next year.

TABLE OF CONTENTS

NOTATION

SYMBOL	DEFINITION
$\underline{A}$	Vector area
$\underline{a},\underline{b},\underline{c}$,	Vectors joining vertices of a finite-difference element
e	Specific internal energy
h	Specific enthalpy
$\underline{k}$	Unit vector in z direction
$\underline{n}$	Unit outer normal
p	Static pressure
q	Flow speed, $q^2=u^2+v^2+w^2$
r	Radial coordinate
$r_H(x), r_S(x)$	Hub, shroud radius
t	Time
u,v,w	Velocity components in x,y,z directions
x,y,z	Streamwise, normal, spanwise coordinates
$\underline{x}$	Position vector
Γ	Circulation
ρ	Density
Ω	Coordinate system angular velocity

<u>Subscripts</u>

i,j,k	Mesh point indices
I	Inlet
E	Exit
U	Upper mesh boundary
L	Lower mesh boundary

1. INTRODUCTION

Modern-day design of turbomachinery depends strongly upon developing an understanding of complicated flow phenomena in rotor passages including viscous-inviscid interactions and fully three-dimensional effects. At present, no theoretical tool exists which accurately and efficiently describes such phenomena. Designers must depend upon inviscid theories which generally require empirical knowledge of trailing-edge flow deflection angles and rough estimates of effects of key viscous phenomena. Because the required empirical information generally must be gleaned from experimental data and because definitive data are scarce, improved theoretical tools are needed.

Great promise for improved theoretical tools attends recent developments in inviscid axial turbomachine throughflow theory, in numerical simulation methods, and in three-dimensional turbulent boundary-layer theory. Considering first improvements in the inviscid theory of axial turbomachines, McCune [1-4] and his co-workers have developed a powerful method for predicting fully three-dimensional flow through axial turbomachinery. The method offers great promise as a design tool as it applies to highly-loaded blading and accounts for nontrivial effects of three dimensionality. The theory, however, is inapplicable within the rotor passages and, to date, leaves shocks unaccounted. Advances in numerical methods, particularly those of MacCormack [5] and Hung, [6] show that numerical simulation of flow in the rotor passages is quite feasible, particularly under neglect of viscous effects. In the particular context of turbomachines, Thompkins and Epstein [7] have successfully computed inviscid flow through a transonic compressor rotor. Hung's computations show that, for three-dimensional external flows, computing time for a viscous simulation will exceed that of an inviscid simulation by as little as 50%, including the presence of a significant separated region. The third advance has been in the field of turbulent boundary-layer theory. The ability to accurately predict properties of three-dimensional boundary layers is steadily improving. The ability to compute such properties on an arbitrary wing at all flow speeds has been developed by Cebeci, et al.[8] In a recent study,[9] the advanced turbulence models developed by Wilcox and Traci[10,11] and by Wilcox and Rubesin[12] have been incorporated in the Cebeci program. Predictions are encouraging. Synthesis of these three methods potentially will yield the most advanced treatment of axial turbomachine flow yet developed.

This study has been undertaken in order to accomplish such a synthesis. In this report we discuss progress made during the initial year of research and projected work to be accomplished during the next year. Because so many novel features are being included in the computer program that will become the desired computational tool for axial turbomachines, the program is not yet fully operational. Hence, no simulations are included. Rather, we present details of the underlying theory.

2. WORK ACCOMPLISHED TO DATE

The primary objective of this research project is to develop an efficient and accurate computational procedure for predicting viscous flow through axial turbomachinery. Such a procedure will be developed by first developing an inviscid three-dimensional computational tool and then incorporating viscous effects through either (a) coupling with a three-dimensional boundary-layer program for attached regions and full Navier-Stokes computational methods on separated regions; or, (b) full Navier-Stokes methods throughout. In the first year of this project our focus has been on the inviscid phase of the problem. We have made important progress toward completing development of a three-dimensional, inviscid computer program which allows for arbitrary distribution of mesh points. Details follow:

2.1 OUTLINE OF THE OVERALL APPROACH

In developing a three-dimensional inviscid program we have chosen to begin with a program developed by MacCormack and Paullay[13] of the NASA Ames Research Center. The program has been designed for computation of transonic viscous flow past arbitrary airfoil sections and permits arbitrary distribution of finite-difference mesh points. Additionally, the program has an automatic mesh-generation option which yields a mesh contoured to the airfoil. Through a remesh option, the program handles shock waves by automatically aligning the mesh with the shock and by locally applying the Rankine-Hugoniot relations. To render this program applicable for three-dimensional inviscid flows the following sequence of steps are being taken:

1. Incorporate three-dimensional input/output logic, suppress viscous-flow provisions and add end planes.
2. Incorporate provision for three-dimensional geometry.

3. Revise equations of motion to account for computing in an axisymmetric, rotating coordinate system.

4. Revise mesh-generation method to provide consistency with periodicity boundary conditions.

5. Incorporate McCune's[2] theory to provide downstream boundary conditions.

6. Generalize the procedure for handling shocks to accommodate three dimensionality.

To date the first four steps have been completed and Step 5 is in progress. We have thus developed an inviscid three-dimensional program suitable for computing flows over arbitrary wings (with the limitation that the program treats shocks in a less-rigorous manner than in the original 2-D version until Step 6 is completed). In the following subsections, we present details of the equations of *motion*, *geometry* provisions embodied in the program, and boundary conditions.

2.2 EQUATIONS OF MOTION

In constructing the program, the equations of motion are written in a cartesian coordinate system and effects of axisymmetry are included in the geometry package. In order to write the equations of motion we let x, y and z denote rectangular cartesian coordinates oriented in the streamwise, blade normal, and spanwise directions (Figure 1).

The coordinate system rotates at angular velocity Ω about the x axis. Velocity components in the x, y and z directions are denoted by u, v and w, respectively. Letting t, ρ, p, e and h denote time, density, static pressure, specific internal energy, and specific enthalpy, the equations for conservation of mass, momentum and energy for inviscid three-dimensional flow are:

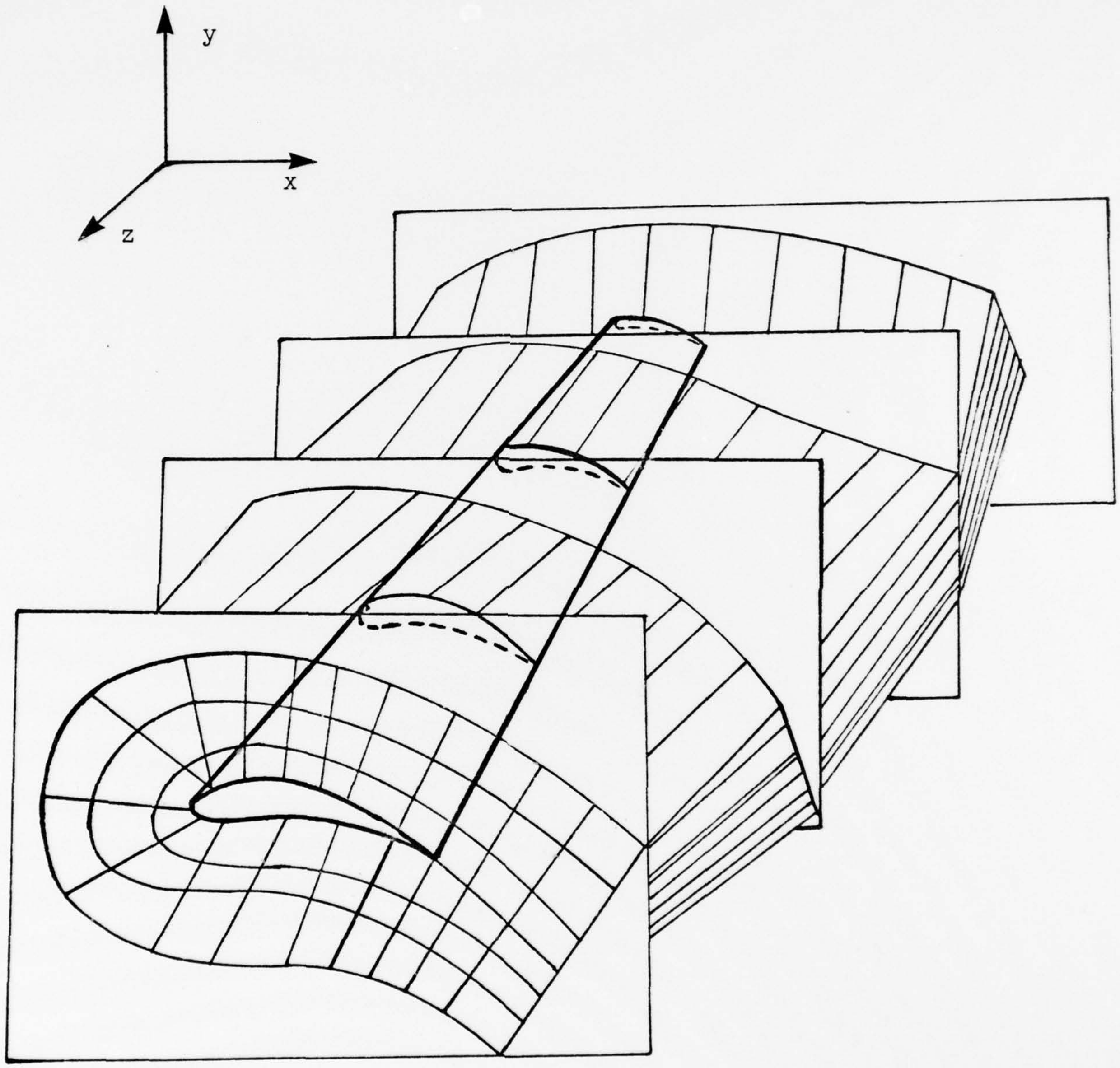


Figure 1. Schematic of a section of the finite-difference mesh.

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho u) + \frac{\partial}{\partial x}(\rho u^2 + p) + \frac{\partial}{\partial y}(\rho vu) + \frac{\partial}{\partial z}(\rho wu) = 0 \tag{2}$$

$$\frac{\partial}{\partial t}(\rho v) + \frac{\partial}{\partial x}(\rho uv) + \frac{\partial}{\partial y}(\rho v^2 + p) + \frac{\partial}{\partial z}(\rho wv) = 2\rho\Omega u + \rho\Omega^2 y \tag{3}$$

$$\frac{\partial}{\partial t}(\rho w) + \frac{\partial}{\partial x}(\rho uw) + \frac{\partial}{\partial y}(\rho vw) + \frac{\partial}{\partial z}(\rho w^2 + p) = -2\rho\Omega v + \rho\Omega^2 z \tag{4}$$

$$\frac{\partial}{\partial t}\left[\rho\left(e + \frac{1}{2}q^2\right)\right] + \frac{\partial}{\partial x}\left[\rho u\left(h + \frac{1}{2}q^2\right)\right] + \frac{\partial}{\partial y}\left[\rho v\left(h + \frac{1}{2}q^2\right)\right] + \frac{\partial}{\partial z}\left[\rho w\left(h + \frac{1}{2}q^2\right)\right] = 0 \tag{5}$$

where

$$q^2 = u^2 + v^2 + w^2 \tag{6}$$

Equations (1) - (6) will be solved using the MacCormack time-splitting method. As complete details of the method have been presented elsewhere,[5,6,13] for the sake of brevity we present no details here.

2.3 GEOMETRIC DETAILS

Figure 1 schematically depicts a section of a three-dimensional mesh about a blade. For simplicity, in the spanwise direction parallel planes are shown to pass normal to the wing. There is no such restriction in the program, however, as such a limitation would preclude flow simulations with varying hub and/or shroud radius. In these planes, which we shall refer to as "k planes," lies a "wrap-around" mesh contoured to the local blade section. Mesh-point distribution is completely arbitrary.

In solving the equations of motion, it is necessary to compute the surface areas and the volume of a given finite-difference element. The element with which we are working is a hexahedron (see Figure 2). Computing the vector areas $\underline{A}_{i+1}$, $\underline{A}_{j+1}$, $\underline{A}_{k+1}$ (and similarly those of the opposite faces, viz, $\underline{A}_i$, $\underline{A}_j$ and $\underline{A}_k$) can be accomplished by breaking each side into two triangles and by using vector algebra to compute the area of each triangle. (Note that conceptually we <u>must</u> do this if the vertices of a given face are not coplanar.) Focusing upon $\underline{A}_{k+1}$ we refer now to Figure 3. Mesh coordinates for the four vertices are written compactly in terms of the position vectors $\underline{x}_{i,j,k}$ at each vertex. The vector area of a triangle is given by half the cross product of vectors joining a common vertex and the remaining two vertices, i.e., denoting the vector areas of triangles I and II by $\underline{A}_I$ and $\underline{A}_{II}$ we have:

$$\left.\begin{aligned} \underline{A}_I &= \tfrac{1}{2}\, \underline{a} \times \underline{b} \\ \underline{A}_{II} &= \tfrac{1}{2}\, \underline{b} \times \underline{c} \end{aligned}\right\} \quad (7)$$

where

$$\left.\begin{aligned} \underline{a} &\equiv \underline{x}_{i+1,j,k+1} - \underline{x}_{i,j,k+1} \\ \underline{b} &\equiv \underline{x}_{i+1,j+1,k+1} - \underline{x}_{i,j,k+1} \\ \underline{c} &\equiv \underline{x}_{i,j+1,k+1} - \underline{x}_{i,j,k+1} \end{aligned}\right\} \quad (8)$$

Using the prescription given in Equation (8) insures that, as required in the finite-difference formulation, the area vector will always be an outer facing normal on the k+1 face. If for simplicity we assume all four vertices lie in the same xy plane and we redenote the vertices as 5, 6, 7, 8, rather than (i,j,k+1), (i+1,j,k+1), (i+1,j+1,k+1), (i,j+1,k+1) as in Figures 2 and 3, Equations (7) simplify to:

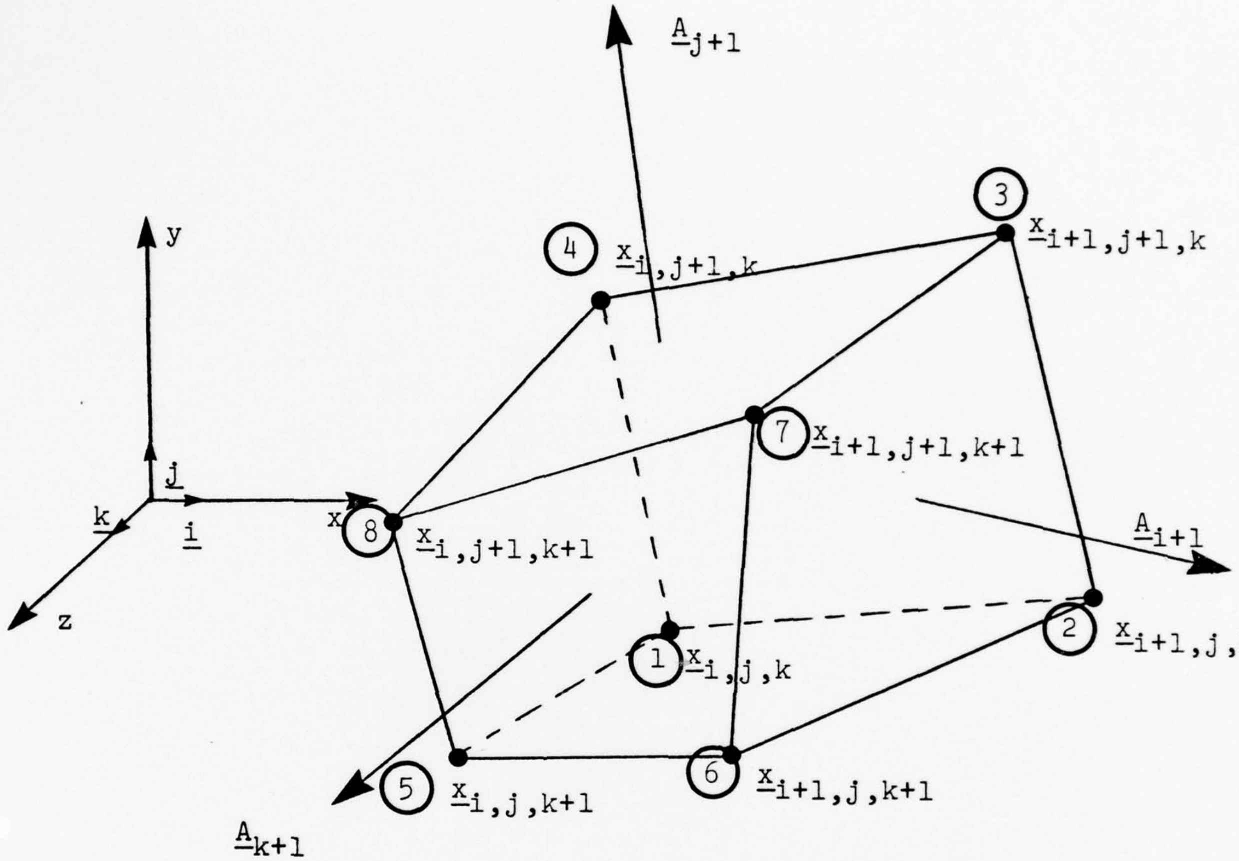


Figure 2. Typical hexahedral finite-difference element.

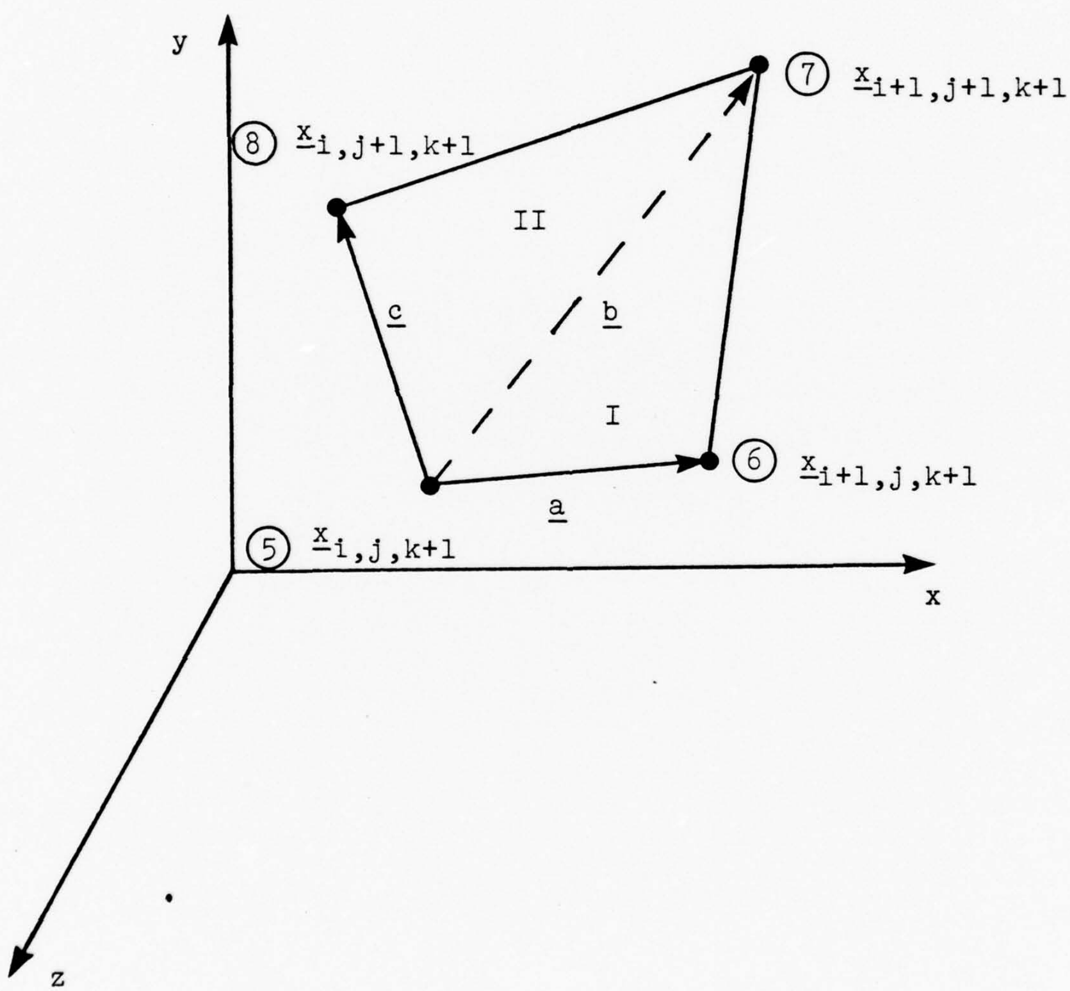


Figure 3. Geometry of the k+1 face.

$$\left.\begin{aligned} \underline{A}_I &= 1/2 \left[(x_6-x_5)(y_7-y_5) - (x_7-x_5)(y_6-y_5)\right] \underline{k} \\ \underline{A}_{II} &= 1/2 \left[(x_7-x_5)(y_8-y_5) - (x_8-x_5)(y_7-y_5)\right] \underline{k} \end{aligned}\right\} \quad (9)$$

where $\underline{k}$ is a unit vector in the z direction.

Computing the volume is also conveniently done by vector algebra, plus knowledge of two key solid-geometry theorems. First, any hexahedron can be broken up into five tetrahedra. Second, the volume of a tetrahedron is one-third the product of its base area and its height (altitude). Consider the tetrahedron whose vertices are points 1, 2, 3 and 6 (see Figure 4). Let $\underline{a}$ and $\underline{b}$ denote the vectors lying along the sides joining vertices 1 and 2 and 3, respectively, i.e.,

$$\left.\begin{aligned} \underline{a} &= \underline{x}_{i+1,j,k} - \underline{x}_{i,j,k} \\ \underline{b} &= \underline{x}_{i+1,j+1,k} - \underline{x}_{i,j,k} \end{aligned}\right\} \quad (10)$$

Then, the base area becomes

$$\underline{A}_B = \frac{1}{2}\, \underline{a} x \underline{b} \quad (11)$$

The altitude can be computed as the dot product of the unit normal to the plane of $\underline{a}$ and $\underline{b}$ with the vector $\underline{c}$ joining vertices 1 and 6. The normal, $\underline{n}$, is given by:

$$\underline{n} = \frac{\underline{a} x \underline{b}}{|\underline{a} x \underline{b}|} \quad (12)$$

where $|\underline{a} x \underline{b}|$ denotes the magnitude of $\underline{a} x \underline{b}$. Hence, the volume becomes

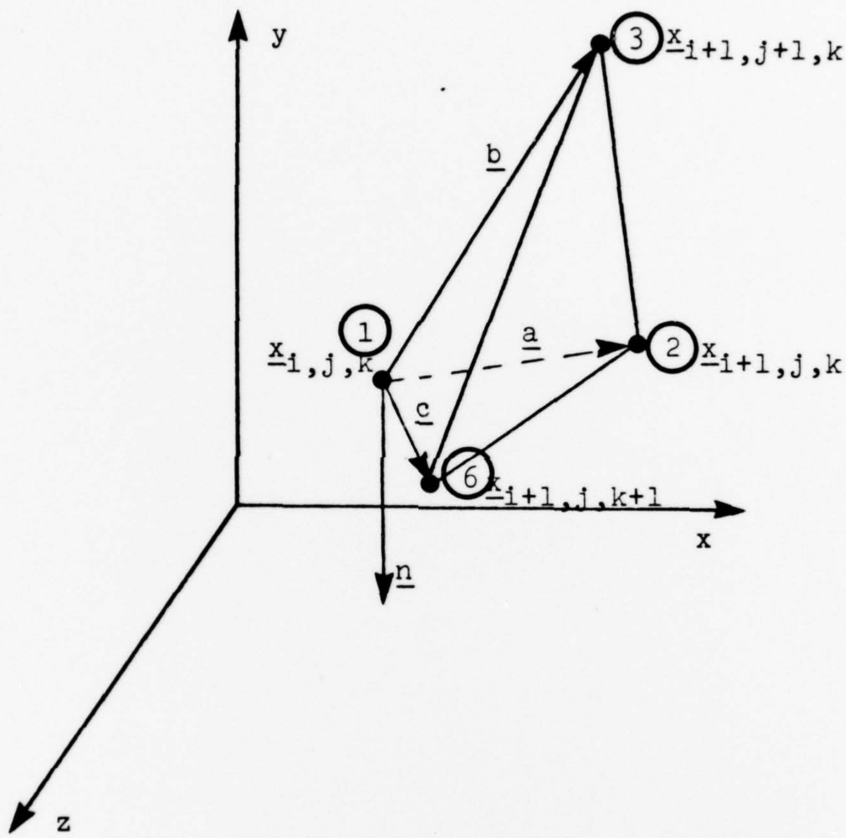


Figure 4. Typical tetrahedron; the vector $\underline{n}$ is normal to the plane of the vectors $\underline{a}$ and $\underline{b}$.

$$
\begin{aligned}
V_{1236} &= 1/3\ |1/2\underline{a}\mathrm{x}\underline{b}|\ \underline{n}\cdot\underline{c} \\
&= 1/6\ \underline{A}_B\cdot\underline{c} \\
&= 1/6\ \underline{a}\mathrm{x}\underline{b}\cdot\underline{c}
\end{aligned}
\tag{13}
$$

In terms of previous notation, we have finally,

$$
V_{1236} = 1/6(\tilde{x}_2\underline{i}+\tilde{y}_2\underline{j}+\tilde{z}_2\underline{k})\ \mathrm{x}\ (\tilde{x}_3\underline{i}+\tilde{y}_3\underline{j}+\tilde{z}_3\underline{k})\cdot(\tilde{x}_6\underline{i}+\tilde{y}_6 j+\tilde{z}_6\underline{k})
\tag{14}
$$

where $\tilde{x}_i \equiv x_i - x_1$, etc. (15)

In applying Equation (14) the vertex numbering system given in Figure 2 is quite important. The key feature of the numbering system is that (a) on adjacent planes the numbering proceeds in the same direction, i.e., counterclockwise and (b) vertices at which one starts (e.g., 1 and 5 for k planes) are adjacent. The volume of the original hexahedron is given by the sum of the volumes of the five tetrahedra, viz,

$$
V = V_{1236} + V_{1348} + V_{1568} + V_{3678} + V_{1368}
\tag{16}
$$

Equation (16) is applicable to arbitrary hexahedra.

2.4 BOUNDARY CONDITIONS

The most unique aspect of the computational procedure follows from the boundary conditions which must be somehow specified or generated at the mesh boundaries as discussed below. While the boundary conditions at the hub and shroud are standard (i.e., vanishing normal fluid flux), those at the other mesh boundaries are much more novel and complex. Figure 5 shows a cross-sectional plane on a k plane; the surfaces I and E are the inlet and exit boundaries respectively while the boundaries denoted by U and L are symmetrically located between adjacent blades. Note that the two blades beyond the boundaries are shown only to help

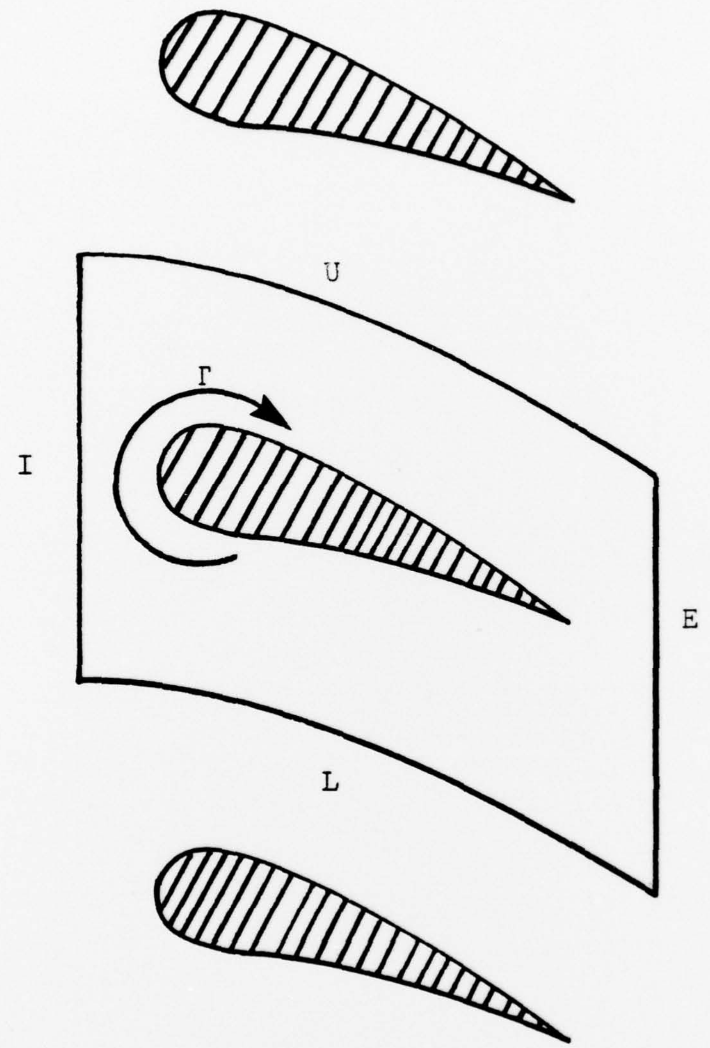


Figure 5. Schematic of computational mesh in a plane of constant radius.

illustrate the geometry and are not included in the computational domain. The quantity Γ is the (instantaneous) circulation about the blade section.

On boundaries U and L we impose periodicity conditions upon the solution. Hence, for a given flow property, ϕ, we require that at, say, each axial position

$$\phi_U = \phi_L \tag{17}$$

where subscript denotes boundary. The boundaries U and L must, of course, be of identical shape and must be symmetrically located between adjacent blades. While it would be desirable for these boundaries to lie in stream surfaces, their locations are not of critical importance to the solution method.

On the one hand, the use of periodic boundary conditions is nothing new for internal flow studies, although their use has implications which may not be generally obvious from experience in external-flow calculations. On the other hand, the boundary conditions we propose using at the inlet and exit boundaries are new and should prove quite innovative and almost certainly will add great economy to our computational procedure.

The boundary conditions we intend to use are based on a recently developed external flow theory for highly-loaded axial turbomachines[1-3, 14, 15] and will be implemented as follows:

1. At the start of the computation, input data will be specified including blade geometry, inlet conditions from far upstream (e.g., inlet flow, angular momentum and enthalpy), and an initial guess at the internal flowfield (including circulation, Γ, about blade sections). These input

data do not include fluxes, separately, across either inlet or exit surfaces (I and E).

2. From the far upstream inlet conditions and the instantaneous Γ, the external analytical theory will be used to determine flow properties along the surfaces I and E, including the necessary surface fluxes. The information required by the external theory to do this is merely the difference in fluxes at "corresponding" points on I and E.

3. The manner in which these flux differences will be generated from the internal flowfield (either from the initial guess or from later iterates) is first to trace out "instantaneous streamlines" corresponding to that flow as if it were steady. Using these connection streamtubes, and appropriate conservation laws, the required net fluxes can be obtained.

4. Starting from the initial guess (or preceding iterate) and using the surface fluxes determined in Steps 2 and 3, the internal computation will proceed for a number of timesteps (the number to be determined from experience) thus improving, or iterating on the flowfield in Step 1.

5. Steps 2 through 4 will be repeated until a converged solution has been obtained.

There are several key points about this procedure which are worthy of note. The most important reason for adopting such a method is to insure solution accuracy. Just as having far-field boundary conditions which reflect conservation of circulation is critically important for accurate numeral computation of flow about an isolated airfoil, so are realistic inlet- and exit-surface boundary conditions (no matter where they are placed) absolutely essential for accurate computation of flow through a rotor passage. While the need for accurate specification of exit-surface conditions will come as no surprise to the experienced numerical fluidflow mechanicist, the corresponding need for the inlet surface may require further examination. The need becomes apparent if one notes that even in a transonic rotor the axial Mach number is generally less than unity so that any effect downstream can in principle be felt in the upstream region. This effect is further exaggerated by the fact that we are working with rotating flows which are attended by significant coupling between pressure changes and vorticity perturbations[16, 17] (such as those produced by various types of wakes in swirling flows[3]). Consequently great care must be taken to account for solution perturbations on the inlet surface, even when that surface is relatively far from the blade.

A second key point is the inherent efficiency attending the procedure because, thanks to the accuracy of McCune's method, the inlet and exit surfaces can be placed within a tenth of a chordlength of the blade. In so doing, we are able to confine most of the mesh points to the immediate vicinity of the blade with an obvious saving in computing time relative to a computation requiring inlet and exit surfaces displaced several chordlengths from the blade.

As an important observation about the overall procedure, note that while our procedure actually solves the "analysis problem" we actually use McCune's theory in the "design-problem" mode. On the one hand, given blade geometry and inlet conditions, our objective is to determine the machine's performance. One of the key quantities we seek in solving the flowfield is the circulation, Γ, a quantity which evolves in time until the steady-state solution is attained. On the other hand, we are using the instantaneous circulation in addition to the inlet conditions to determine flow properties on the inlet and exit planes via McCune's theory. This is, by definition, the classical design problem. Now there are two parts of the design-problem solution which, until steady-flow conditions have been achieved, will be inconsistent with the instantaneous internal solution, viz, the line on which the blade wake intersects the exit surface and the blade shape. The latter point is a conceptualization only and is of no actual consequence to the solution procedure as McCune's theory will not be used in the internal region where it could in principle generate a blade shape consistent with the instantaneous Γ. The former point is of key importance, however, and may serve as an important measure of the rate of approach to steady-flow conditions.

3. FUTURE WORK

During the next year the inviscid version of the program will be debugged and tested for an axial turbomachine test case to be selected. In this section, we briefly describe the main thrust of our projected research efforts after the initial debugging has been accomplished.

3.1 MESH-GENERATION PROCEDURE FOR ARBITRARY THREE-DIMENSIONAL GEOMETRIES

An integral aspect of our overall approach is to avoid working in nonphysical coordinates. By contrast we prefer to use rectangular cartesian coordinates and confine all geometric irregularities to the geometry package. That is, we will reflect the axisymmetry and any variable hub and/or shroud radius complications through computation of finite-difference element vector areas and volumes.

To explain our projected procedure, we consider Figure 6 which shows a view of a rotor passage in an xz plane. The hub and shroud radii are denoted by $r_H(x)$ and $r_S(x)$, respectively where the radial coordinate, , is related to the cartesian coordinates by

$$r^2 = y^2 + z^2 \tag{18}$$

Our procedure for generating the overall finite-difference mesh will consist of the following six steps:

1. Generate blade-surface points along a z=constant k surface. This will be done by interpolation from input blade-section data.

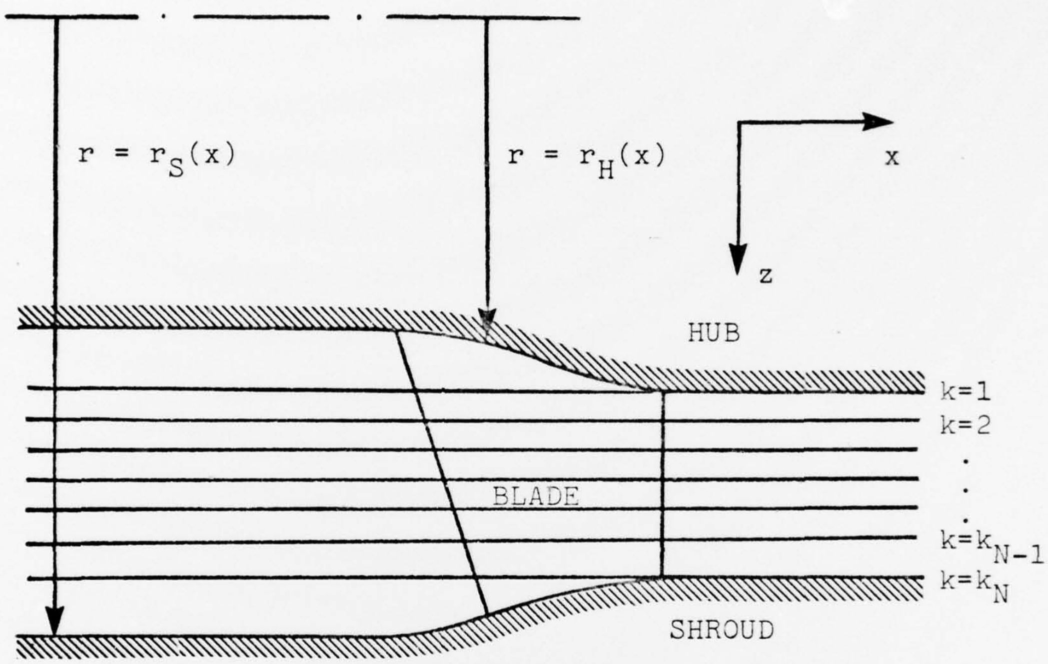


Figure 6. Top view of a rotor passage including initial locations of k surfaces. After deformation of the k surfaces, k=1 lies along the hub, $k=k_N$ lies along the shroud, and the other surfaces are deformed by linear interpolation.

2. Deform the z surface to conform with blade-passage geometry. This will be done by linear interpolation. That is, if there are ten surfaces between hub and shroud, the nth surface from the hub, measuring from the hub, will lie n-tenths of the local distance between hub and shroud.

3. By interpolation, find new blade-surface points along the deformed k surface. Again the input blade-section data will be used.

4. Generate complete x-y mesh for the deformed blade section determined in Step 3. This step uses the mesh-generation procedure already embodied in the MacCormack-Paullay[13] program.

5. Compute local values of z for the x-y mesh generated in Step 4. Note that in all preceding steps only the values of z on the blade surface are computed. In this step we are filling out the rest of the mesh.

6. Project the y,z coordinates onto cylindrical surfaces. To do this we compute

$$\left.\begin{aligned} y &= \frac{\hat{y}\hat{z}}{\sqrt{\hat{y}^2+\hat{z}^2}} \\ z &= \frac{\hat{z}^2}{\sqrt{\hat{y}^2+\hat{z}^2}} \end{aligned}\right\} \tag{19}$$

where y and z denote the values of $\hat{y}$ and $\hat{z}$ computed in Step 5.

Using this procedure precludes having to transform the equations of motion into a nonphysical coordinate system. Consequently, there will be no need to compute local metrics. We, in effect, compensate by computing local area vectors and volumes.

3.2 PROPER HANDLING OF SHOCK WAVES

To properly handle shock waves we plan to generalize the remesh option of the MacCormack-Paullay program for three dimensions. In this option the mesh is altered from time to time in order to align a finite-difference mesh line (surface for three dimensions) with the shock.

3.3 INCORPORATION OF VISCOUS EFFECTS

Originally we had planned to tie a three-dimensional boundary-layer program into the overall program in order to compute boundary-layer properties. This would leave a requirement to find some other computational procedure for separated regions, probably a full Navier-Stokes computation. It now appears evident that such a hybrid scheme is impractical and that a full Navier-Stokes computation is more sensible. This is particularly attractive as MacCormack has made such improvements to his time-splitting method that a full viscous computation can be done with as little as a 50% computing-time increase over that required for an inviscid computation. Hence, we plan to devise a full Navier-Stokes program as our end product.

4. DISCUSSION

In summary, we have made important progress toward developing a computer program which can be used to predict three-dimensional flow in a rotating turbomachine, including treatment of viscous effects. This is being done in two steps. In the first step we are developing an inviscid program. In the second step viscous effects and more rigorous treatment of shocks will be added. The debugging phase for the inviscid program is nearing completion.

REFERENCES

1. McCune, J.E. and Hawthorne, W.R., "The Effects of Trailing Vorticity on the Flow Through Highly-Loaded Cascades," JFM, Vol 74, p 721 (Apr 1976).

2. McCune, J.E. "Three-Dimensional Inviscid Flow Through a Highly-Loaded Transonic Compressor Rotor," paper presented at Workshop on Transonic Flow Problems in Turbomachinery, Monterey CA (Feb 1976).

3. Tan, C.S., "Three-Dimensional Vorticity Induced Flow Effects in Highly-Loaded Axial Compressors," MIT PhD Thesis (Jun 1978).

4. Chen, L.T. and McCune, J.E., "Comparison of Three-Dimensional Quasi-Linear Large Swirl Theory with Measured Outflow from a High-Work Compressor Rotor," MIT Gas Turbine Lab Report No 128 (Sep 1975).

5. MacCormack, R.W., "An Efficient Numerical Method for Solving the Time-Dependent Compressible Navier-Stokes Equations at High Reynolds Number," NASA TM X-73,129 (Jul 1976)

6. Hung, C.M., "Numerical Simulation of Three-Dimensional Shock Wave and Boundary-Layer Interaction," DCW Industries Report DCW-R-09-02(Sep 1977).

7. Thompkins, W.T., Jr. and Epstein, A.H., "A Comparison of the Computed and Experimental Three Dimensional Flow in a Transonic Compressor Rotor," AIAA Paper 76-368 (Jul 1976).

8. Cebeci, T., Kaups, K., Ramsey, J. and Moser, A., "Calculation of Three-Dimensional Compressible Boundary Layers on Arbitrary Wings," Aerodynamic Analyses Requiring Advanced Computers, NASA SP-347, pp 41-76 (Mar 1975).

9. Wilcox, D.C., "Computation of Three-Dimensional Boundary-Layers with a Two-Equation Turbulence Model," DCW-R-11-03 (Jun 1978).

10. Wilcox, D.C. and Traci, R.M., "A Complete Model of Turbulence," AIAA Paper 76-351 (Jul 1976).

11. Wilcox, D.C., "A Model for Transitional Flows," AIAA Paper 77-126 (Jan 1977).

12. Wilcox, D.C. and Rubesin, M.W., "Progress in Turbulence Modeling for Complex Flowfields," DCW Industries Report DCW-R-11-01 (Apr 1977).

13. MacCormack, R.W. and Paullay, A., "Computational Efficiency Achieved by Time Splitting of Finite Difference Operators," AIAA Paper 72-154 (January 1972).

14. Adebayo, A.O., "Three-Dimensional Beltrami Flow Through a Highly-Loaded Axial Compressor Rotor Inducing Arbitrary Swirl," MIT PhD Thesis (Jun 1978).

15. McCune, J.E., "Three-Dimensional Flow in Highly-Loaded Axial Turbomachines," MS No 56180 (22S2), ZAMP (Nov 1977).

16. Greenspan, H.P., The Theory of Rotating Fluids, Cambridge Univ Press (1968).

17. Kerrebrock, J.L., "Small Disturbances in Turbomachine Annuli with Swirl," AIAA Journal, Vol 15, No 6, pp 794-803 (Jun 1977).